Available online at www.sciencedirect.com
ScienceDirect
Procedia Technology 25 (2016) 892 – 899
Procedia Technology

Global Colloquium in Recent Advancement and Effectual Researches in Engineering, Science and Technology (RAEREST 2016)

Computational study of hypersonic flow past spiked blunt body using RANS and DSMC method

Jiss J Sebastian[a], Sandeep Eldho James[a], Abhilash Suryan[a]*

[a]*College of Engineering Trivandrum-695016, Kerala, India*

Abstract

Hypersonic vehicles when moving at very high speeds experience the problem of drag and heating. One of the ways to reduce this drag and heating is by the use of an aerospike. In the present study, the flow around a blunted body fitted with an aerospike is analyzed using a commercial software ANSYS Fluent and an open source Direct Simulation Monte Carlo (DSMC) code, called as dsmcFoam in OpenFOAM, at a high Mach number (M=6) at different length to diameter ratios (L/D = 1.5, 2) at an angle of attack 0°. The aerospike placed in front of the body replaces the strong detached shock wave ahead of the body with a system of weaker oblique shock waves. A recirculation region is developed between the shock and the blunt body, which acts like a streamlined profile, thus reducing the drag and wall heat flux.

Peer-review under responsibility of the organizing committee of RAEREST 2016

Keywords: Aerospike; Hypersonic flow; Recirculation zone;RANS; DSMC method

1. Introduction

For deceleration and thermal management, vehicles like space planes, reusable launch vehicles, missiles and interplanetary probes which fly at supersonic and hypersonic speeds usually employ blunt nosed bodies. The reduction of both drag and aerodynamic heating is a major challenge in designing such vehicles. In the context of high speed hypersonic vehicles, blunt geometries are preferred over slender ones as the former accounts for higher volumetric

* Corresponding author. Tel.: +91-828-121-1932.
E-mail address: suryan@cet.ac.in

Peer-review under responsibility of the organizing committee of RAEREST 2016
doi:10.1016/j.protcy.2016.08.174

efficiency, effective accommodation of crew or on-board equipment [1]. A blunt nosed body creates a bow shock wave ahead of it at high Mach numbers. An aerospike fitted in front of the blunt nose helps in reducing drag. Flow past the aerospike creates a conical shock wave, replacing the bow shock, which will be far away from the body [2]. Flow behind the shock wave separates on the aerospike and a recirculation region is created near the body surface.

Nomenclature

C_p	pressure coefficient
C_{pg}	specific heat of air at constant pressure, J/(kg-K)
D	cylindrical forebody diameter, m
DF	drag force taken in horizontal direction, N
d	aerospike diameter, m
E	energy, J/kg
G_k	generation of turbulence kinetic energy due to the mean velocity gradients
G_b	generation of turbulence kinetic energy due to buoyancy
g	acceleration due to gravity, m/s^2
h	sensible enthalpy
I	unit tensor
k	turbulent kinetic energy, J/kg
k_{eff}	effective thermal conductivity, W/(m-K)
k_g	thermal conductivity of air, W/(m-K)
L	length of aerospike, m
M	Mach number
P_0	stagnation pressure, N/m^2
Pr_t	turbulent Prandtl number
p	pressure, N/m^2
p_{ref}	reference pressure, N/m^2
q_{ref}	reference dynamic pressure, N/m^2
R	cylindrical forebody radius, m
Re	flow Reynolds number
S	reference area based on the cylinder diameter, m^2
s	arc length, m
T	temperature, K
T_0	stagnation temperature, K
v_{ref}	reference velocity, m/s
Y_M	contribution of fluctuating dilatation in the compressible turbulence to overall dissipation rate
α	angle of attack, deg
ε	dissipation rate, J/(kg-s)
μ	dynamic viscosity, kg/(m-s)
μ_{eff}	effective dynamic viscosity, kg/(m-s)
μ_t	turbulent dynamic viscosity, kg/(m-s)
ρ	density, kg/m^3
σ_k	turbulent Prandtl number for k
σ_ε	turbulent Prandtl number for ε

In the present study, high speed flow at a Mach number, M = 6, past a blunt body fitted with aerospike of different spike length to blunt diameter (L/D) ratios at 0^o angle of attack are studied numerically using different software ANSYS Fluent and dsmcFoam. The numerical predictions are compared with corresponding experimental results.

2. Literature review

Alexander at the Langley pilotless aircraft division is said to be the first to suggest aerospike for drag reduction on blunt bodies at supersonic speeds [3]. Study performed by Mair is said to be the first landmark study in this field [4]. He studied experimentally the flow around spiked flat cylindrical and hemisphere cylindrical models at a Mach number of 1.96 and Reynolds number of $1.65x10^5$. He recorded a flow instability around the spiked bodies and proposed an explanation for this flow oscillation based on the pressure difference between the flow downstream of the reattachment shock and the flow inside the recirculation zone. The label 'spike' was used first by Piland and Putland [5]. They concluded that no drag reduction was achieved for transonic range at Reynolds numbers from $1.44x10^5$ to $2.64x10^5$ flight conditions regardless of the aerospike lengths. Jones [6] studied that the drag reduction depend on both rod length and aerodisk geometry. For longer rods, the aerodisk was found to lose its benefits in reducing the drag.

It is seen that, majority of the investigations regarding the use of aerospike or aerodisks are limited to the assessment of drag reduction abilities. Stalder and Nielsen [7] carried out the first study on the aerothermodynamic effects of the spikes and they measured the heat transfer to a hemisphere cylindrical model for Mach numbers ranging from 0.12 to 5.04. They concluded that regardless of the length of aerospike and tip geometry, heat transfer to the spiked model was as twice as that of the unspiked model. They found the reason for the heat transfer rise to be the high turbulence level in the dead air zone and impingement of turbulent shear layer on the model surface. Crawford performed an experimental study of drag and aerodynamic heating on a spiked hemisphere cylindrical model in Mach 6.8 flow [8]. The model's surface pressure and heat flux attained peak values at the reattachment point. Yamauchi et al. [9] studied a numerical simulation of the flow field around an aerospike fitted blunt body at free stream Mach numbers of 2.01, 4.14 and 6.80 for different values of L/D. Holden studied that the peak local heat transfer rate at the reattachment point is in a direct proportionality with the reattachment angle [10]. Khlebnikov inferred that the value of peak heat flux varies inversely with its distance from the root of aerospike [11].

The studies done by Gerdroodbary et al. [12] shows a reduction in the total heat transfer rate by employing hemispherical aerodisks in turbulent flow conditions. The shortest aerospike should be avoided as this length may lead to the same impingement location of bow shock and the reattachment shock. Myshenkov [13] applied the solution of complete Navier-Stokes equations using finite-difference approach in two dimensional (2D) axisymmetric computational domain. Paskonov et al. [14] solved the complete unsteady Navier-Stokes equations in a 2-D axisymmetric flowfield around cone cylindrical and flat cylindrical models equipped with pointed aerospikes of length to diameter ratios varying up to 1.Subsequently, a large number of investigations were carried out to understand the effect of high speed flows past a blunt body with a projecting aerospike at the tip. Majority of the investigations conducted for spiked blunt bodies' have concluded that the use of aerospikes can drastically reduce the aerodynamic drag at very high speeds viz., supersonic and hypersonic speeds, for certain ratios of length of aerospike to the diameter of the base body.

Studies done by Motoyama et al. [15], Milicev and Pavlovic [16], Menezes et al. [17, 18], Kalimuthu et al. [19] and Mehta et al. [20, 21] showed that the effectiveness of the aerospikes can be increased further by the use of flat faced or hemispherical faced spikes called the aerodisks. Modifying the tip of a plain aerospike was proved to have a strong impact on its performance even without using an aerodisk. It was inferred by Milicev and Pavlovic that the aerospike with a rounded tip produces the maximum drag reduction.

Yadav et al. [22] investigated the effect of employing two aerodisks in series on the peak reattachment heat flux and drag for a hemisphere cylinder. Their results showed that a proper aerodisk-aerospike design with two aerospikes in series has an ability of reducing the aerodynamic heating of a re-entry vehicle even under fully turbulent conditions. Tahani et al. studied different geometries of aerodisks and aerospikes [23]. They concluded that the cut spike delivers the lowest level reduction in drag and an aerodisk is more effective than the aerospike. Recent advances in the field of aero-thermodynamics of spiked hypersonic vehicles are detailed by Ahmed and Qin [24].

3. Numerical model

3.1. Governing Equations

Numerical simulations were obtained for analyzing a hypersonic flow past blunt body fitted with aerospike of different length to diameter ratios. The present analysis is steady and hence, the governing equations are solved in its steady form. These system of equations are closed with the ideal gas equation of state. Governing equations used by ANSYS Fluent [25] for the numerical analysis are given in general form as follows:

Continuity equation:

$$\partial\rho/\partial t + \nabla.(\rho v) = 0 \tag{1}$$

Momentum equation:

$$\partial(\rho v)/\partial t + \nabla.(\rho v v) = -\nabla p + \nabla.\mu[(\nabla v) - 2/3\,\nabla.vI] \tag{2}$$

Energy equation:

$$\partial(\rho E)/\partial t + \nabla.(v(\rho E + p)) = \nabla.(k_{eff}\,\nabla T + (\mu_{eff}\,[(\nabla v) \; - 2/3\,\nabla.vI].v) \tag{3}$$

Where, p is the static pressure, ρ is the density, μ is the molecular viscosity, I is the unit tensor, k_{eff} is the effective conductivity, ρE is the total energy per unit volume, T is the temperature. In Eq. (3), $E = h - (p/\rho) + (1/2)\,v^2 k_{eff} = k_g + k_t$, the effective thermal conductivity, where $k_t = (C_{pg}\mu_t)/Pr_t$, Pr_t is turbulent Prandtl number. Last term in the energy equation represents viscous heating and $\mu_{eff} = \mu + \mu_t$.

3.2. Turbulence modeling

Two equation standard k-ε model proposed by Launder and Spalding [26] is used in the present study. The model is characterized by robustness, economy and reasonable accuracy for a broad range of turbulent flows. The model was selected on the basis of the extensive literature survey and studies conducted using different turbulence models viz., one equation Spalart-Allmaras model, two equation standard k-ε model, two equation RNG k-ε model and two equation k-ω SST model. It is based on the model equations of turbulent kinetic energy (k) and dissipation rate (ε) as shown below.

$$\partial(\rho k)/\partial t + \partial(\rho k u_i)/(\partial x_i) = \partial/(\partial x_j)\,[(\mu + \mu_t/\sigma_k)\;\partial k/(\partial x_j)] + G_k + G_b - \rho\varepsilon - Y_M \tag{4}$$

$$\partial(\rho\varepsilon)/\partial t + \partial(\rho\varepsilon u_i)/(\partial x_i) = \partial/(\partial x_j)\,[(\mu + \mu_t/\sigma_\varepsilon)\;\partial\varepsilon/(\partial x_j)] + C_{1\varepsilon}\;\varepsilon/k\,(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\,\varepsilon^2/k \tag{5}$$

Where G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants which have the values 1.44, 1.92 and 0.09 respectively. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, which are assigned the values 1 and 1.3 respectively.

3.3. DSMC method

Direct Simulation Monte Carlo (DSMC) method is a physical simulation technique commonly used for solving high Knudsen number flows. The region of gas flow is simulated by a large number of representative particles. All of the molecular alternations such as movements and collisions are computed. In DSMC method, molecular motion and surface interactions are simulated using deterministic approach and collisions using probabilistic approach. The main procedure followed involves particle initialization, movement of particle and boundary interaction, intermolecular collisions and macroscopic averaging in the cells for field variant and in the wall surface for surface pressure and heat flux variant. An open source Direct Simulation Monte Carlo code, called as dsmcFoam in OpenFOAM is employed for solving the present problem. Air is used in the present analysis with fractions of nitrogen (N_2) and oxygen (O_2). MaxwellianThermal wall interaction model is used. Binary Collision model used is LarsenBorgnakkeVariableHardSphere and the value of the relaxationCollisionNumber is taken as 5. InflowBoundaryModel used is FreeStream. The dimension of cell and time step has a significant role on the accuracy of result. So this value must be such that particles in a step time move less than the mean free path of the molecules. Hence the size of cell must be one third of the calculated mean free path. The dsmc particles in the present analysis is approximately 5 lakhs.

3.4. Geometry and grid

The geometrical dimensions of the spiked blunt body used in this study are shown in Fig. 1 [27]. The model is two dimensional, with a cylindrical forebody having a hemispherical nose and an aerospike attached to the stagnation point of the nose cone. The diameter of the cylindrical forebody, D, is taken as 40 mm and the length of this body as 1.25D. The end of the aerospike is taken as hemispherical with a diameter 0.1D as shown in the Fig. 1. The length to diameter ratio is taken as 1.5 and 2 depending on the case selection.

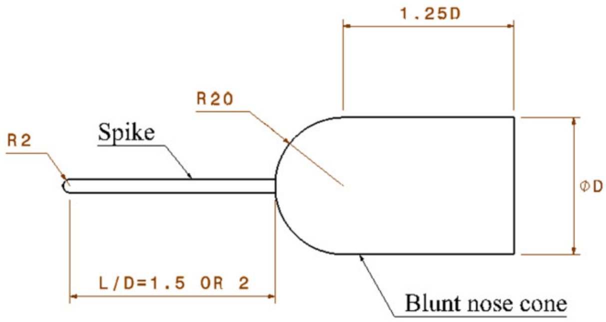

Fig. 1. Geometry of the model.

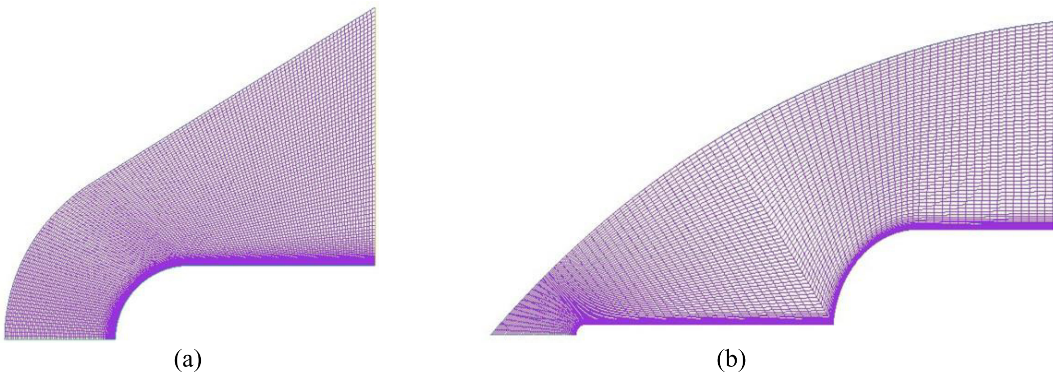

Fig. 2. Grid generated (a) without spike; (b) L/D 1.5.

Free stream conditions are assigned to the inlet and outlet of the domain while stationary wall condition is applied to the surface of the spiked blunt body [27]. Free stream inflow conditions at a Mach number 6 were used for inlet boundary. Wall temperature is assigned as 300 K. Values of the free stream flow are the stagnation pressure, P_0 = 8.3 bar, stagnation temperature, T_0 = 450 K, and a flow Reynolds number, Re = 5 x 10^5 based on the spike length, L = 60 mm. Mesh is created using ANSYS ICEM CFD. Block mesh strategy is used. The mesh developed for no-spike case and L/D ratio 1.5 case are shown in Fig. 2.

4. Results

Results of the RANS and DSMC numerical simulations carried out shows that a strong bow shock is generated at a small distance from the spiked blunt body. Flow behind the shock wave separates on the spike and a recirculation region is created near the body surface. As a result of this recirculation, the pressure and wall heat flux decrease in the region ahead of the blunt body. However, the reattachment of the shear layer on the shoulder of the blunt body increases the values of local heat flux and pressure. The pressure on the surface of the body can be significantly reduced when the aerospike replaces the bow shock with a system of weaker oblique shock waves. Mach number variation of flow over a blunt body without aerospike using dsmcFoam and ANSYS Fluent is shown in Fig. 4(a). A bow shock is formed in front of the body. This shock is better captured using DSMC method when comparing with ANSYS Fluent. Figure 4(b) shows the Mach number variation of flow over a blunt body equipped with an aerospike of L/D ratio 1.5. A strong bow shock is developed in front of the body and can be visualized from the figures. Shock wave is best captured in dsmcFoam. Figure 4(c) shows the Mach number variation of flow over a blunt body equipped with an aerospike of L/D ratio 2. The flow separation as a result of using aerospike can be visualized from the numerical results. The lower values of temperature in the recirculation region cools the blunt body surface. The size of the recirculation region varies for different length to diameter ratios of the aerospike. Since the surface pressure in front of the blunt body is reduced, resultant drag is also reduced. Peak pressure values over the wall for different cases are shown in Table 1.ANSYS Fluent results show that a 20.8% reduction in total pressure can be achieved when using an aerospike of L/D ratio 1.5. In the case of L/D ratio 2, total pressure reduction is 26.9%.Numerical predictions using DSMC shows that by the employment of an aerospike with L/D ratio 1.5 gives 22.4% reduction in total pressure. Aerospike with L/D ratio 2 gives a better reduction when comparing with L/D = 1.5. Total pressure reduction in this case is found to be 27.8%. Total pressure in front of the blunt body for different cases are shown in Table 2.

Table 1. Peak Pressure values over the wall for different cases

	Peak Pressure (Pa)			
	On spike DSMC	On body DSMC	On spike ANSYS	On body ANSYS
No spike	-	24782.20	-	24084.85
L/D 1.5	26853.1	5794.32	26218.44	5799.47
L/D 2	26949.3	4527.58	26343.15	4534.20

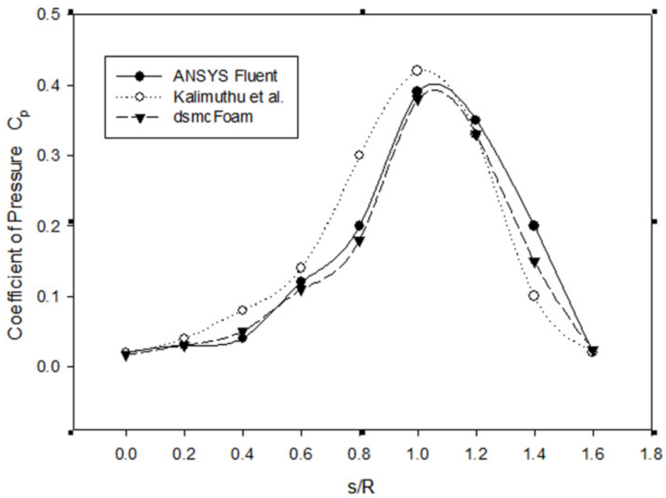

Fig. 3. Coefficient of Pressure on the blunt body vs s/R for L/D 1.5.

Table 2. Total Pressure values ahead of the blunt body for different cases

	Total Pressure ahead of the blunt body (Pa)	
	DSMC	ANSYS
No spike	4177.31	4074.4
L/D 1.5	3412.77	3373.5
L/D 2	3268.92	3211.4

Coefficient of Pressure values predicted by ANSYS Fluent and dsmcFoam are plotted against arc length to radius ratio (s/R) for L.D 1.5 and is compared with the experimental data obtained by Kalimuthu et al. [27] and is shown in the Fig. 3. Coefficient of Pressure is calculated using the following equation.

$$C_p = (p - p_{ref})/(q_{ref}) \quad (6)$$

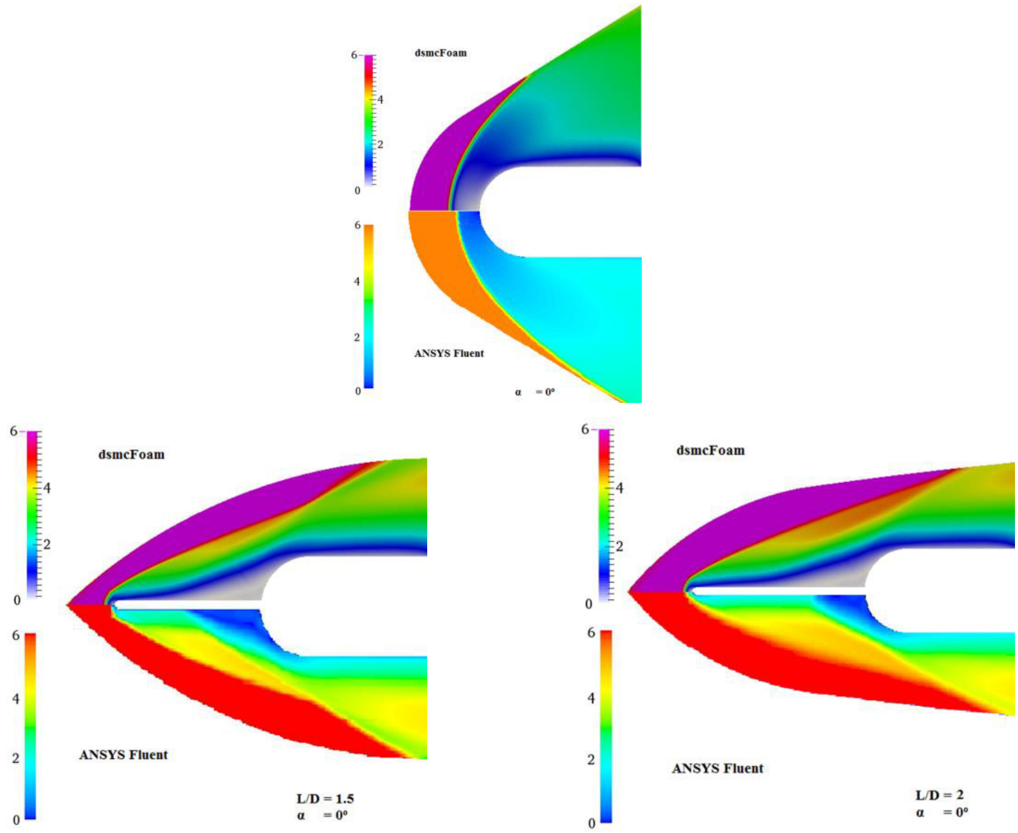

Fig. 4. Mach number contours (a) without spike (top); (b) L/D 1.5 (bottom left); (c) L/D 2 (bottom right).

4. Conclusion

Hypersonic flow visualization at Mach number 6 over an aerospike with varying length to diameter ratio (1.5, 2) and at zero degree angle of attack is numerically simulated using ANSYS Fluent and DSMC method. Pressure variation and Mach number contour are studied. Shock wave is better captured using DSMC method in all cases. Pressure decreases in the stagnation region of the blunt body because of the formation of a recirculation zone. Peak pressure values at spike and blunt body are compared for different cases viz., without spike, L/D 1.5 and L/D 2 using different solvers ANSYS Fluent and DSMC method. Reduction in total pressure ahead of the blunt body is higher for L/D 2 case for both software and a better reduction is predicted using DSMC method. Pressure reduction leads to a reduction in drag. The present computational model is being developed further for studying the effect of various geometries and combinations of drag reduction devices in hypersonic flows. Aerospike tip can be modified and analyzed as it will have an unavoidable effect on the flow field.

References

[1] Shoemaker, J. M., Aerodynamic spike flow fields computed to select optimum configuration at Mach 2.5 with experimental validation, AIAA paper 90-0414, 1990.
[2] Album, H. H., Regarding the utility of spiked blunt bodies, Journal of the Aerospace Sciences, Vol. 5, No. 1, 1968, p. 112–113.
[3] Alexander, S. R., Results of tests to determine the effect of a conical wind shield on the drag of a bluff body at supersonic speeds, NACA RM L6K08a, 1947.
[4] Mair, W. A., Experiments on separation of boundary layers on probes in front of blunt-nosed bodies in a supersonic air stream, Philosophy Magazine, Vol. 43, No. 243, 1952, p. 695–716.
[5] Piland, R. O., Putland, L. W., Zero-lift drag of several conical and blunt nose shapes obtained in free flight at Mach number of 0.7 to 1.3, NACA RM L54A27, 1954.
[6] Jones, J. J., Experimental drag coefficients of round noses with conical wind- shields at Mach Number 2.72, NACA RM L55E10, 1955.
[7] Stalder, J. R., Nielson, H. V., Heat transfer from a hemisphere cylinder equipped with flow separation spikes, NACA TN 3287, 1954.
[8] Crawford, D. H., Investigations of the flow over a spiked nose hemisphere cylinder, NASA TN-D-118, 1959.
[9] Yamauchi, M., Fujii, K., Higashino, F., Numerical investigation of supersonic flows around a spiked blunt body, Journal of Spacecraft and Rockets, Vol. 32, No. 1, 1995, p. 32-42.
[10] Holden, M., Experimental studies of separated flows at hypersonic speeds. Part I-separated flows over axisymmetric spiked bodies, AIAA Journal, Vol. 4, No. 4, 1966, p. 591-599.
[11] Khlebnikov, V. S., Distribution of heat flux on the surface of a sphere with a forward separation zone, Fluid Dynamics, Vol. 21, No. 4, 1986, p. 654-658.
[12] Gerdroodbary, M. B., Hosseinalipour, S. M., Numerical simulation of hypersonic flow over highly blunted cones with spike, Acta Astronautica, Vol. 67, 2010, p. 180-193.
[13] Myshenkov, V. N., Numerical investigation of separated flow in front of a spiked cylinder, Fluid Dynamics, Vol. 16, No. 6, 1981, p. 938–942.
[14] Paskonov, V. M., Cheranova, N. A., Numerical investigation of laminar separation in the case of supersonic flow of viscous gas past spiked bodies, Fluid Dynamics, Vol. 19, No.2, 1984, p. 281– 285.
[15] Motoyama, N., Mihara, K., Miyajima, R., Watanuki, T., Kubota, H., Thermal protection and drag reduction with use of spike in hypersonic flow, AIAA paper, 2001.
[16] Milicev, S. S., Pavlovic, M. D., Ristic, S., Vitic, A., On the influence of spike shape at supersonic flow past blunt bodies, Mechanics, Automatic Control and Robotics, Vol. 3, No. 12, 2002, p. 371-382.
[17] Menezes, V., Saravanan, S., Jagadeesh, G., Reddy, K. P. J., Experimental investigations of hypersonic flow over highly blunted cones with aerospikes, AIAA, Vol. 41, No. 10, 2003, p. 1955–1966.
[18] Menezes, V., Kumar, S., Maruta, K., Reddy, K. P. J., Takayama, K., Hypersonic flow over a multi-step afterbody, Shock Waves, 2005, p. 421-424.
[19] Kalimuthu, R., Mehta, R. C., Rathakrishnan, E., Experimental investigation on spiked body in hypersonic flow, The Aeronautical Journal, 2008, p. 593–598.
[20] Mehta, R. C., Numerical analysis of pressure oscillations over axisymmetric spiked blunt bodies at Mach 6.8, Shock Waves, Vol. 11, 2002, p. 431-440.
[21] Mehta, R. C., Kalimuthu, R., Rathakrishnan, E., Flowfield analysis over aero-disc attached to blunt nosed body at Mach 6, Proceedings of the 37th International & 4th National Conference on Fluid Mechanics and Fluid Power, IIT Madras, 2010.
[22] Yadav, R., Velidi, G., Gueven, U., Aerothermodynamics of generic re-entry vehicle with a series of aerospikes at nose, Acta Astronautica, Vol. 96, 2014, p. 1-10.
[23] Tahani, M., Karimi, M. S., Motlagh, A. M., Mirmahdian, S., Numerical investigation of drag and heat reduction in hypersonic spiked blunt bodies, Heat Mass Transfer, Vol. 49, 2013, p. 1369-84.
[24] Ahmed, M. Y. M., Qin, N., Recent advances in the aerothermodynamics of spiked hypersonic vehicles, Progress in Aerospace Sciences, Vol. 47, 2011, p. 425-449.
[25] ANSYS Inc., Fluent user's guide, [USA].
[26] Launder, B. E., and Spalding, D. E., The numerical computation of turbulent flows, Computer methods for applied mechanics and engineering, Vol. 3, 1974, p. 269-289.
[27] Kalimuthu, R., and Rathakrishnan, E., Aerospike in drag reduction for hypersonic flow, AIAA, 2008. p. 92-98